Proceedings of the ASME 2014 Power Conference
Power 2014
July 28-31, 2014, Baltimore, Maryland, USA

Power2014-32157

CFD SIMULATIONS OF FLOW AND HEAT TRANSFER THROUGH THE POROUS INTERFACE OF A METAL FOAM HEAT EXCHANGER

Emilie Sauret*
School of Chemistry, Physics
& Mechanical Engineering
Queensland University of Technology
Brisbane, Queensland, 4000
Australia
Email: emilie.sauret@qut.edu.au

Kamel Hooman
School of Mining
& Mechanical Engineering
The University of Queensland
Brisbane, Queensland, 4072
Australia
Email: k.hooman@uq.edu.au

Suvash C. Saha
School of Chemistry, Physics
& Mechanical Engineering
Queensland University of Technology
Brisbane, Queensland, 4000
Australia
Email: suvash.saha@qut.edu.au

ABSTRACT

This paper offers numerical modelling of a waste heat recovery system. A thin layer of metal foam is attached to a cold plate to absorb heat from hot gases leaving the system. The heat transferred from the exhaust gas is then transferred to a cold liquid flowing in a secondary loop. Two different foam PPI (Pores Per Inch) values are examined over a range of fluid velocities. Numerical results are then compared to both experimental data and theoretical results available in the literature. Challenges in getting the simulation results to match those of the experiments are addressed and discussed in detail. In particular, interface boundary conditions specified between a porous layer and a fluid layer are investigated. While physically one expects much lower fluid velocity in the pores compared to that of free flow, capturing this sharp gradient at the interface can add to the difficulties of numerical simulation. The existing models in the literature are modified by considering the pressure gradient inside and outside the foam. Comparisons against the numerical modelling are presented. Finally, based on experimentally-validated numerical results, thermo-hydraulic performance of foam heat exchangers as waste heat recovery units is discussed with the main goal of reducing the excess pressure drop and maximising the amount of heat that can be recovered from the hot gas stream.

*Address all correspondence to this author.

NOMENCLATURE

c_F	form drag coefficient [-]
dP	pressure drop along the channel [Pa]
$\frac{dP}{dx}$	pressure gradient along the channel in the x-direction $[Pa.m^{-1}]$
H_f	foam height [mm]
H_t	channel total height [mm]
K	permeability $[m^2]$
L	foam length [mm]
P	pressure [Pa]
T	temperature [Pa]
U	mean velocity $[m.s^{-1}]$
W	foam width [mm]
y_w^+	non-dimensional grid spacing at the wall [-]

Greek Symbols

ε	porosity [-]
μ_{eff}	effective dynamic viscosity $[kg.m^{-1}.s^{-1}]$
ρ	density $[kg.m^{-3}]$

Subscripts

out	outlet of the domain
nf	no foam
x	x(flow)-direction
y	y(vertical)-direction
∞	inlet conditions

INTRODUCTION

Metal foam heat exchangers are receiving considerable attention [1–3] especially because of their ability to increase the efficiency of air-cooled heat exchangers for geothermal applications. Metal foams are fibrous materials which are becoming increasingly popular thanks to their attractive thermophysical properties such as high surface-to-volume ratio, low density, thermal and corrosion resistance and high mechanical strength and rigidity [4, 5]. Heat transfer is enhanced by increasing the turbulence and mixing and dispersion induced by the ligaments of the foam as well as by high heat conductivity through the metallic ligaments. These properties lead to smaller, lighter and more efficient heat exchangers which become more attractive than conventional heat exchangers [3, 6, 7].

Challenges in getting the simulation results on metal foam heat exchangers to match those of the experiments consist notably in the interface boundary conditions specified between the porous layer, i.e. the metal foam, and the fluid layer. As recently underlined by Nield and Kuznetsov [8] in their review, the interface modelling remains an open question in the literature. While physically one expects much lower fluid velocity in the pores compared to that of free flow, capturing this sharp gradient at the interface can add to the difficulties of numerical simulation. Beavers and Joseph [9] were amongst the first to show that sharp gradients at the interface between the porous and fluid regions exist. Their experimental work highlighted the existence of a slip velocity at the interface. From there, authors have established different interface conditions that can be classified in to two main types according to Alazmi and Vafai [10]: slip and no-slip boundary conditions. They then establish five main categories for the hydrodynamic interface conditions and four categories for the thermal interface conditions that they critically examine. The different models give mostly comparable results at the exception of few very specific cases. Based on this, the present paper will investigate in details the velocity interface numerically obtained in the case of a metal foam heat exchanger for different PPIs, inlet velocity, and foam heights. It will also offer a comparison with previously published models and discuss the accuracy of the current interface modelling.

COMPUTATIONAL DOMAIN, GRID AND BOUNDARY CONDITIONS

The numerical simulations are performed on a waste heat recovery system.

As shown in Figure 1, a thin layer of metal foam is attached to a cold plate to absorb heat from hot gases leaving the system. The heat transferred from the exhaust gas is then transferred to a cold liquid flowing in a secondary loop. Similarly to Boyd and Hooman [11], the computational domain was chosen longer than the phyiscal one to eliminate the entrance and exit effects. Two different foam layer heights (3 mm and 4 mm) and two different

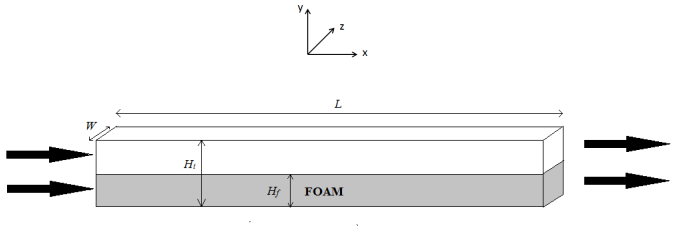

FIGURE 1: SCHEMATIC OF THE COMPUTATIONAL DOMAIN

TABLE 1: METAL FOAM CHARACTERISTICS.

Variables	
K [m^2]	3×10^{-7}, 1×10^{-8}
ε [-]	0.937
PPI	20, 40
H_f [mm]	3, 4
W [mm]	30
L [mm]	195

foam PPI values (20PPI and 40 PPI) are examined over a range of fluid velocity (6 $m.s^{-1}$, 10 $m.s^{-1}$ and 30 $m.s^{-1}$) set at the inlet of the domain. The bottom wall temperature was set constant to 300 K while the inlet temperature was set to 523 K. The top wall was defined with an adiabatic boundary condition.

The metal foam characteristics are given in Table 1.

The two-dimensional computational grid has 23,348 nodes. Results are obtained with a maximum face size of 0.5mm and a minum face size of 0.3mm. It was observed that halving the cell sizes only slightly changed the results. The grid has been refined at the interface between the porous and non-porous regions. At the walls, a 5-layer inflation was defined with a growth rate of 1.2 with the non-dimensional distance at the wall y_w^+ limited to 5. The convergence of the results was carefully checked and all the residuals dropped below 10^{-4}.

NUMERICAL APPROACH

The 2D simulations were carried out using commercially-available software *FLUENT*. The standard k-ε turbulence model was used for the non-porous region following the work of Odabaee et al. [12].

The metal foam domain has been modelled as an isotropic homogeneous porous media using the modified Darcy law (Equation 1)

TABLE 2: COMPARISON OF THE NUMERICAL OUTLET GAS TEMPERATURE FOR A NON-POROUS CHANNEL AND THE NUMERICAL PRESSURE DROP FOR A POROUS CHANNEL.

Inlet Velocity	T_{out} [K] non-porous channel		$dP - dP_{nf}$ [kPa] porous channel	
	Present Simulations	Theory [14]	Present Simulations	Experiments [15]
$U_\infty = 10m.s^{-1}$	467	496	0.85	0.53
$U_\infty = 30m.s^{-1}$	481.7	501	5.3	5.5

$$-\frac{dP}{dx} = \frac{\mu_{eff}}{K}u_x + c_F\frac{\rho}{\sqrt{K}}|U|u_x \quad (1)$$

with c_F = 0.1 for 20PPI and c_F = 0.2 for 40PPI are the experimental values, μ_{eff} the effective viscosity taken equal to the fluid viscosity.
At the interface between the porous domain and the main flow, the continuity in shear stress is applied, similarly to the work presented by Ejlali et al. [13].

RESULTS

Validation against experimental data

The temperature and pressure have been validated in the case of a plain channel without foam and with foam respectivley. The results are presented in Table 2 and are compared against the theoretical and experimental results. The temperature obtained at the outlet of a turbulent pipe is determined using the Dittus-Boelter equation [14] while the pressure difference is established by removing the shear stress from the total pressure as shown by Jadhav et al. [16] and compared against the experimental data from Ackermann [15]. The results show acceptable agreements for the pressure drop at high inlet velocity but a large difference at low inlet velocity. This can be explained by the uncertainties in the experimental data which are expected to be higher at low pressure drop values. Furthermore, the inaccuracy of the current model to predict the interface may also contribute to the difference in pressure drop at low velocity. The gas temperature at the exit of the domain for both inlet velocities is predicted with less than 6% difference compared to the Dittus-Boelter correlation for a non-porous channel. Extensive checks were performed on the simulation and the accuracy for non-porous cases and hal-foamed case showing good agreements with the experimental data.

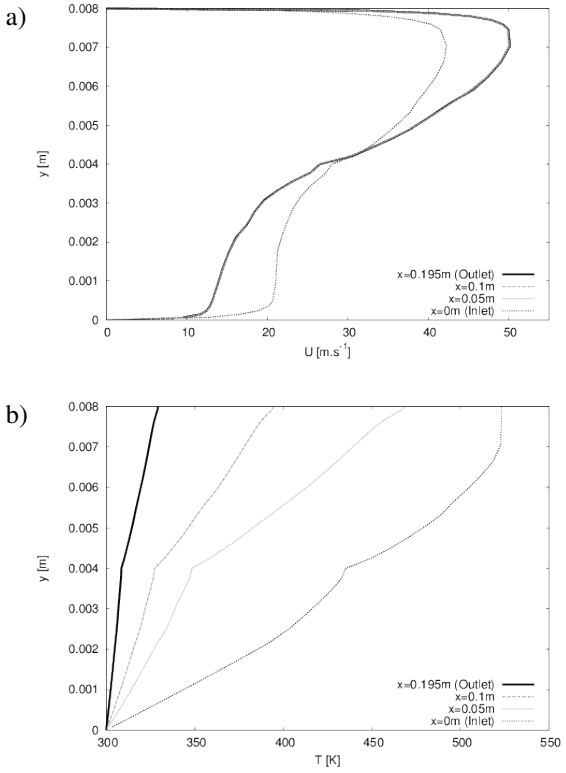

FIGURE 2: a) VELOCITY AND b) TEMPERATURE PROFILES AT 4 LOCATIONS ALONG THE CHANNEL FOR THE 20PPI-4 mm FOAM AT $U_\infty = 30m.s^{-1}$.

The velocity and temperature profiles at different x-locations along the channel are plotted in Figure 2 and the contours are represented Figure 3 for the 20PPI-4mm foam at $U_\infty = 30m.s^{-1}$. As expected, the gas temperature keeps changing along the channel and asymptotically approaches the cold wall temperature at the exit of the domain. However, the air velocity is fully developped before the flow travels a quarter of the channel length. The flow development in porous channels is known to be very rapid, within 10 hydraulic diameter from the inlet for turbulent pipe flows. The CFD simulations achieve it within 6 channel height.
Similar results are obtained with the 3-mm foam height.

Effect of PPI

One can note that there are almost no effects of the PPI on the velocity profiles as shown in Figure 4. The influence on the temperature profiles is also small (Figure 4) with a decrease of the maximum temperature at the exit of 5 degrees.

The normalized pressure drop and the thermal resistance evolutions with the PPI are presented in Figure 5. As expected, the increase in PPI leads to an increase of pressure drop for all the inlet velocities associated with a decrease of the thermal resistance. However, the decrease of thermal resistance is almost

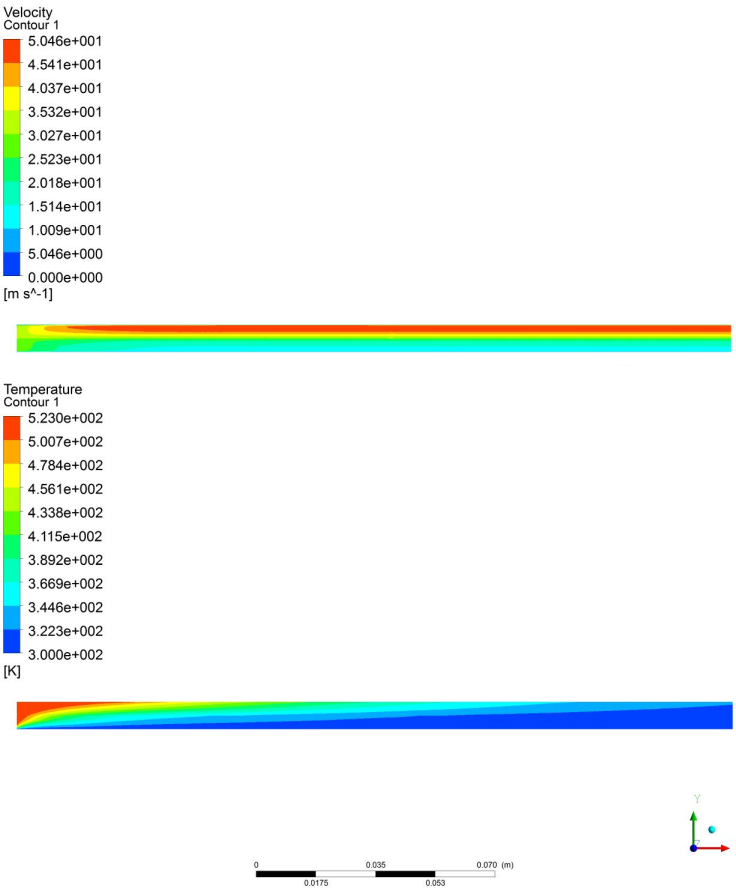

FIGURE 3: VELOCITY AND TEMPERATURE CONTOURS FOR THE 20PPI-4 mm FOAM AT $U_\infty = 30m.s^{-1}$.

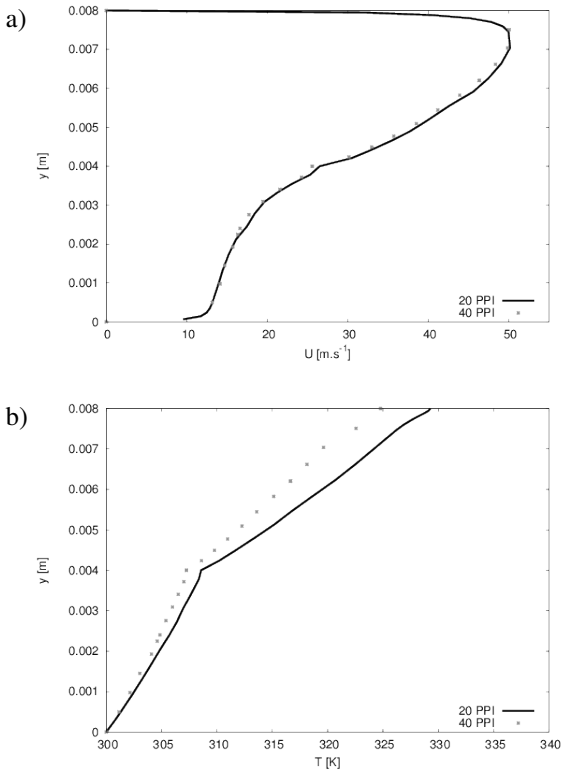

FIGURE 4: a) VELOCITY AND b) TEMPERATURE PROFILES AT THE EXIT OF THE DOMAIN FOR 20PPI AND 40PPI FOR THE 4 mm FOAM HEIGHT AT $U_\infty = 30m.s^{-1}$.

negligible at the highest inlet velocity. The increase of the pressure drop for all inlet velocities is also almost negligible with a maximum increase of 8% at the lowest inlet velocity.

Comparison between the present model and the theoretical interface model of Kuznetsov

The numerical velocity profiles are compared against the jump conditions proposed by Kuznetsov [17] at the interface.

One can note on the comparisons between the numerical and theoretical velocity profiles presented in Figure 6 is that the CFD is in relatively good agreement with the theory. The theoretical profiles from Kuznetsov have been obtained using a two numerical pressure drops, i.e. the pressure drop inside the porous media and the pressure drop in the non-porous region. This allows a more accurate agreement between the CFD and the theoretical profiles. The first difference between the numerical velocity profiles and the theoretical profiles as shown in Figure 6 is the maximum velocity in the porous region. The theory gives as maximum velocity the inlet velocity while the CFD has a lower velocity which is expected due to the foam slowing down the flow inside the porous media. As the theoretical profiles are established based on a laminar flow in the non-porous region, the velocity profiles tend to have the maximum velocity at the center of the non-porous region. Because the numerical simulations are turbulent, the maximum velocity is at the outlet of the boundary layer. However, the maximum velocity in the clear region correlates well with Kuznetsov's theory. Finally, Beavers and Joseph [9] showed that sharp gradients at the interface between the porous and fluid regions exist. The Beavers-Joseph condition can be written as [18]:

$$\frac{\partial u}{\partial y}_{|y=0(interface)} = \frac{\alpha}{\sqrt{K}}(u_{slip} - U_D) \qquad (2)$$

where u_{slip} is the slip velocity at the interface, U_D, the Darcy velocity inside the porous domain, α, the slip coefficient and K the permeability.

The slip coefficient has been shown to strongly depend on

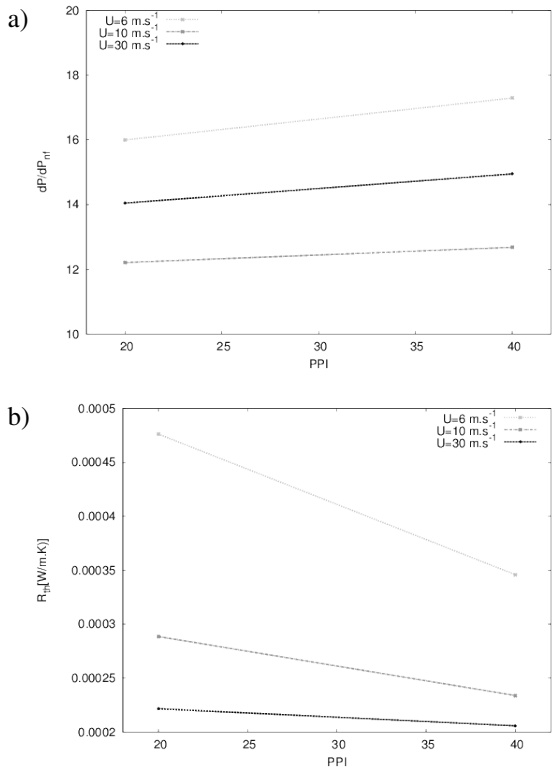

FIGURE 5: EVOLUTION OF THE a) PRESSURE DROP AND b) THERMAL RESISTANCE WITH THE PPI FOR THE 4 mm FOAM HEIGHT AT THE THREE INLET VELOCITIES.

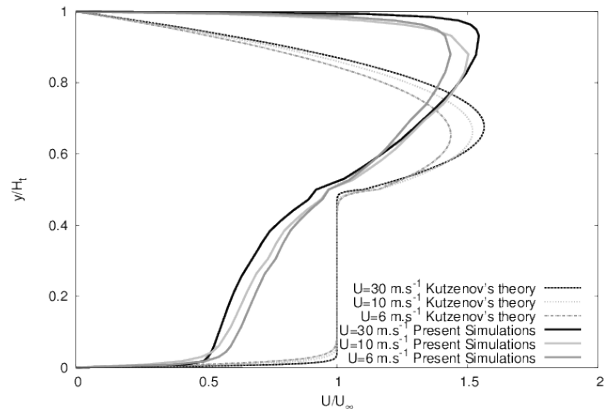

FIGURE 6: COMPARISON OF THE CFD AND THEORETICAL NON-DIMENSIONAL VELOCITY PROFILES AT THE OUTLET OF THE DOMAIN FOR THE 20PPI-4 mm FOAM AT DIFFERENT INLET VELOCITIES.

the surface structure and material properties of the foam such as the porosity [18] [19].
Kuznetsov [17] also sets a stress-jump condition at the interface to model these gradients. Because of a continuous shear stress condition between the porous and non-porous regions, the numerical simulations don't show any sharp gradients at the interface. In order to take this effect into account in our model, a slip condition at the interface for the mass and momentum equations may lead to improved results.

CONCLUSION

This paper offered numerical modelling of a waste heat recovery system with two different foam PPI values over a range of fluid velocities. The interface boundary conditions specified between a porous layer and a fluid layer are investigated showing as one expects a much lower fluid velocity in the pores compared to that of free flow. The existing models in the literature are modified by considering the pressure gradient inside and outside the foam which gives acceptable comparisons with the numerical simulations. However, the numerical model assumes a continuous shear stress condition between the porous and non-porous regions and so can't capture the sharp gradients at the interface.

Further investigation will look at modifying the numerical interface conditions with a slip condition.

REFERENCES

[1] Boomsma, K., Poulikakos, D., and F., Z., 2003. "Metal foams as compact high performance heat exchangers". *Mechanics of MAterials,* ***35***, pp. 1161–1176.

[2] Ejlali, A., Ejlali, A., Hooman, K., and Gurgenci, H., 2009. "Application of high porosity metal foams as air-cooled heat exchangers to high heat load removal systems". *International Communications in Heat and Mass Transfer,* ***36***(7), pp. 674–679.

[3] Odabaee, M., and Hooman, K., 2012. "Metal foam heat exchangers for heat transfer augmentation from a tube bank". *Applied Thermal Engineering,* ***36***(1), pp. 456–463.

[4] Mahjoob, S., and Vafai, K., 2008. "A synthetis of fluid and thermal transport models for metal foam heat exchangers". *International Journal of Heat and Mass Transfer,* ***51***, pp. 3701–3711.

[5] T'Joen, C., De Jaeger, P., Huisseune, H., Van Herzeele, S., Vorst, N., and De Paepe, M., 2010. "Thermo-hydraulic study of a single row heat exchanger consisting of metal foam covered round tubes". *International Journal of Heat and Mass Transfer,* ***53***, pp. 3262–3274.

[6] Calmidi, V., and Mahajan, R., 2000. "Forced convection in high porosity metal foams". *Journal of Heat Transfer,* ***122***, pp. 557–565.

[7] Odabaee, M., De Paepe, M., De Jaeger, P., T'Joen, C., and Hooman, K., 2013. "Particle deposition effects on heat transfer from a metal foam-wrapped tube bundle". *Inter-*

national Journal of Numerical Methods for Heat and Fluid Flow, ***23***(1), pp. 74–87.

[8] Nield, D., and Kuznetsov, A. V., 2013. "An historical and topical note on convection in porous media". *Journal of Heat Transfer,* ***135***, pp. 061201–1–061201–10.

[9] Beavers, G., and Joseph, D., 1967. "Boundary conditions at a naturally permeable wall". *Journal of Fluid Mechanics,* ***30***, pp. 197–207.

[10] Alazmi, B., and Vafai, K., 2001. "Analysis of fluid flow and heat transfer interfacial conditions between a porous medium and a fluid layer". *International Journal of Heat and Mass Transfer,* ***44***, pp. 1735–1749.

[11] Boyd, B., and Hooman, K., 2012. "Air-cooled micro-porous heat exchangers for thermal management of fuel cells". *International Communications in Heat and Mass Transfer,* ***39***(3), pp. 363–367.

[12] Odabaee, M., Hooman, K., and Gurgenci, H., 2011. "Metal foam heat exchangers for heat transfer augmentation from a cylinder in cross-flow". *Transport in Porous Media,* ***86***(3), pp. 911–923.

[13] Ejlali, A., Mee, D., Hooman, K., and Beamish, B., 2011. "Numerical modelling of the self-heating process of a wet porous medium". *International Journal of Heat and Mass Transfer,* ***54***(25-26), pp. 5200–5206.

[14] Incropera, F. P., and DeWitt, D. P., 2007. *Fundamentals of Heat and Mass Transfer*. New-York: Wiley.

[15] Ackermann, D., 2012. "Experimental investigation of fouling in various coolers". PhD thesis, University of Stuttgart.

[16] Jadhav, P., V., R., and Hooman, K., 2013. "Experimental analysis of metal foam filled double pipe heat exchangers". In 8th World Conference on Experimental Heat Transfer, Fluid Mechanics, and Thermodynamics, June 16-20, Lisbon, Portugal.

[17] Kuznetsov, A., 2000. *Handbook of Porous Media - Chapter 6: Analytical Studies of Frorced Convection in Partly Porous Configurations*. CRC Press, New-York.

[18] Chandesris, M., and Jamet, D., 2006. "Boundary conditions at a planar fluid?porous interface for a poiseuille flow". *International Journal of Heat and Mass Transfer,* ***49***, pp. 2137–2150.

[19] Sahraoui, M., and Kaviany, M., 1992. "Slip and no-slip velocity boundary conditions at interface of porbus, plain - media". *International Journal Heat Mass Transfer,* ***35 (4)***, pp. 927–943.